

## Abstract

Most of previous analysis had assumed the flow in ejectors as a simple one-dimensional compressible flow. Such the flow omits the existence of shocks in the flow which is normally induced by the primary fluid entering the ejector at very high speed and hence cannot be used to represent the real flow phenomena inside the ejector. The aims of this study are to investigate the use of CFD in predicting performance of an R141b ejector used in refrigeration application and to reveal the complications of the flow characteristics reflected to the performance of the ejector. The performances of the ejector were evaluated in term of the entrainment ratio ( $R_m$ ), and the critical back pressure ( $P_C$ ). In order to obtain the accurate models, the CFD results were validated with the experimental values. In this study, an experimental R141b ejector refrigerator with the test ejector was built. The test ejector was designed to be easily fitted with different pieces so that the investigation on the effect of geometries would be allowed. It was found that the predicted performances of the simulated models were agreed well with the experimental values. Average errors of the predicted entrainment ratio and the critical back pressure were 9% and 2%, respectively.

After the validations were satisfied, the changes in the flow phenomena inside the R141b ejector, when its operating conditions and geometries were varied, could be analyzed. Using the applications provided by the CFD software, the flow structure of the modeled ejectors could be created graphically, and the phenomena inside the flow passage were explored. Introducing a new parameter, primary flow state (the state of the primary flow immediately after leaving the primary nozzle), which is a combined parameter of the ejector's operating conditions, it was discovered that when the ejector operated with various primary flow states, this parameter had a major impact reflected to the flow and the mixing process in the ejector.

In addition to the effects of operating conditions, the effects of geometry's variations including the primary nozzle throat diameter, the mixing camber inlet diameter, the throat length, and the primary nozzle exit position (NXP) were also investigated. The investigation on the effect of the ejector's geometries on the flow characteristics and the performance of the ejector were made. The simulation results show that the performances of the ejector varied when there was a change in the geometries. However, the change in the performance when the geometries were changed also depended on the primary flow state. At the different primary flow states, the change in the ejector's geometries could affect the change in the flow in different ways.

To be concluded, the CFD was found to be not only a sufficient tool in predicting ejector performance, it also provides a better understanding of the flow and mixing processes within the ejector. Significant phenomena of the flow in the ejector, such as choke flow, mixing behavior, jet core effect and the presence of oblique shock, which cannot be investigated in the one-dimensional analyses, were explored using useful functions available in the CFD.